Prediction and mitigation of turbulence by CFD modelling of a turbine intake channel at a Small Hydro Plant. Counter-intuitive outcomes and hydraulic common sense

Cristian Fregoni
Frosio Next S.r.l.
Via Corfù, 71
25124 Brescia
Italy

Luigi Lorenzo Papetti
Frosio Next S.r.l.
Via Corfù, 71
25124 Brescia
Italy

Intro

This paper presents the results of a set of multiphase three-dimensional CFD simulations, performed for a turbine intake channel of a small run-of-river hydropower plant in Northern Italy. The aim of the study was to determine, thanks to CFD simulations, the flow field inside the designed intake channel, identify possible recirculation areas which could extend downstream affecting the turbine efficiency and, if needed, carry out the hydraulic optimization of the plant layout.

The complexity of such flow structures, often not predictable by insight neither describable by a simplified 2D model, makes the design of the intake channels a big challenge, especially if the plant layout needs to respect local constraints and a compromise needs to be found. Evidence of such phenomena in existing plants incentivized the plant owner investing on a CFD simulation.

In the case study, a set of preliminary simulations helped get confidence with the turbulent structures forming inside the domain, resulting in a higher control and reliability of the results. The final simulation of the designed channel revealed some criticisms, therefore, an hydraulic optimization of the plant layout was needed. In particular, the combined effect of the sudden horizontal contraction of the channel and the inclined bottom resulted in the formation of a complex three-dimensional turbulent structure which propagates downstream and reaches, only at superficial layers, the turbine entrance. Two different layout alternatives have been investigated with the intent of finding the configuration which best mitigate the formation of such turbulent structures: the increase of the radius of curvature in correspondence of the contraction of the channel, the opening of the environmental flow rate gate located on the weir. Both the solutions affected positively the behaviour of the flow and their results will be presented and discussed in the paper.

1. Background

The main objective of the hydropower intakes design is to mitigate phenomena such as submerged vortices, free-surface vortices, non-uniform spatial distribution of the velocity at the impeller and the presence of gas bubbles that can decrease drastically the turbine efficiency and therefore the energy production of the hydropower plant. Resulting phenomena such as cavitation and vibration inevitably lead to a reduction of the turbine operational life and subsequently an increase of the maintenance costs.

Such complex flow fields, hardly describable by a simplified 2D model and often not predictable by insight, can be well determined by a sophisticated technique such as a CFD (Computational Fluid Dynamics) simulation.

2. The case study

The hydraulic functioning of the designed plant is typical of side-off run of river plants: the river is diverted, by means of a weir, into a lateral channel which conveys the water flow towards the turbine; the water is then returned to the river. The designed channel is placed on orographic left side and a sub-horizontal axis Kaplan turbine is installed. The maximum exploitation flow rate is 26.00 m^3/s and the water level within the intake channel is maintained, thanks to the regulation of the turbine distributor opening, at the constant value of 109.57 m a.s.l. (5.58 m from the bottom of the channel). Tab. 1. Summarizes the main data of the plant.

Type of plant	Run of river
Gross head	5.00 m
Maximum flow rate	26.00 m^3/s
Nominal power	~1300 kW

Tab. 1. Concession data of the hydropower plant

The 3D model of the plant is shown in Fig. 1 and includes a portion of the river upstream the barrage, extended for an arbitrary distance (compatible with the boundary conditions of the case), and the turbine canal intake, extended downstream up to the turbine entrance. The layout includes the following peculiarities, which play a considerable role in the possible formation of turbulent structures within the intake channel:

- Curved internal edge: usually provided to soften the negative effects induced by the sudden horizontal contraction of the channel,
- Column at the centre of the intake channel: it is a local constraint of the case,
- Inclined bottom of the intake channel: the turbine intake channel shows a considerable deepening of the bottom needed for hosting the turbine.

Tab. 2 summarizes the geometrical characteristics of the 3D model of the plant.

Main river	*Cross section*	Rectangular
	Width	26 m
	Length	41 m
	Bottom slope	0 %
Turbine channel intake	*Cross section*	Rectangular
	Width	7.5 m
	Length	28 m
	Internal curvature radius	~3 m
	Bottom slope	0.268 %
Structural column	*Length*	6.5 m
	Width	0.5 m

Tab. 2. Geometrical data of the modelled domain

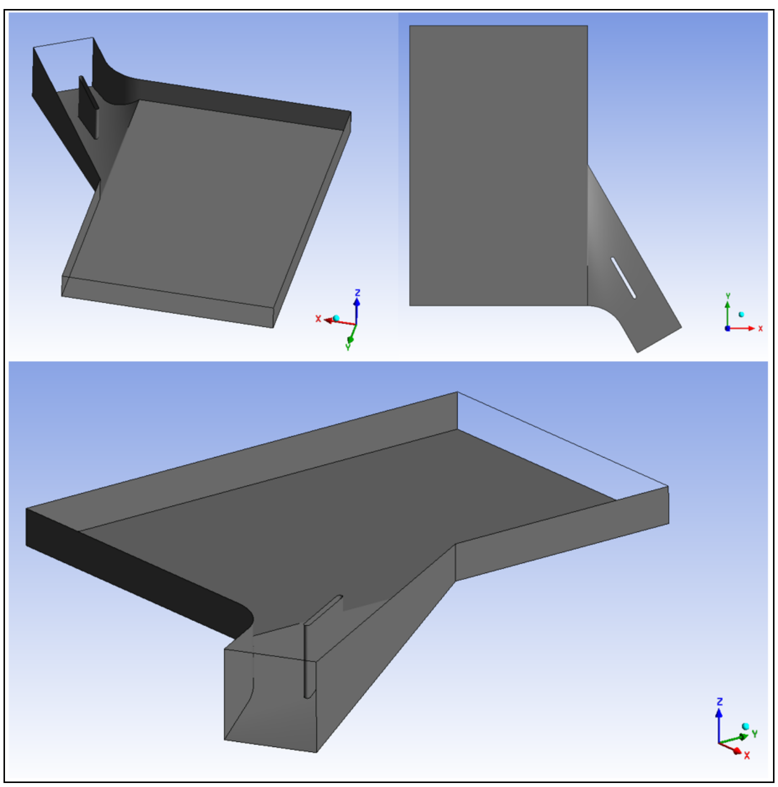

Fig. 1. 3D model of the plant

2.1 Simulations setup

The 3D geometry of the model, shown in Fig. 1, was generated with an integrated drawing software. The domain boundaries were named as follows:

- *Inlet*: upstream section of the river
- *Outlet*: downstream section of the turbine channel intake
- *Environment*: top horizontal face of the domain
- *Walls*: the bottom and the walls

Then, the mesh was generated, by subdividing the domain into ~7 million hexahedral cells. A mesh refinement was applied in the regions where high gradients were expected to occur, hence downstream the channel intake and in correspondence of the surface water level, while along the river section the mesh was maintained coarser in the direction of the flow. The setup of the simulation involved: steady state, multiphase, open channel, VOF (Volume of Fluid), SST k-ω turbulence model. The boundary conditions were specified for each domain surface and basically consisted in setting the flow rate crossing the *inlet* boundary (26.00 m^3/s) and the water level at the *outlet* boundary (regulated by the turbine). A roughness of ks=80 $m^{1/3}/s$ of the concrete structures was assumed. The solution has been initialized with the domain partially full of water in order to be as close as possible to the final solution. The convergence of the solution was checked by monitoring the evolution in time of the residuals (which should fall below 10^{-3}) and the flow rate at the *outlet* (which at the end should equals the flow rate at the *inlet*). The computational time, varying from case to case, was less than 1 hour. A dedicated section of the software was used for the visualization of the results, just in part included in the present paper.

2.2 Step by step approach

A detailed geometry typically involves complex, and sometimes not intuitive, flow structures. The possible risk in these cases is losing the control of the simulation outcomes, resulting in an unreliable CFD simulation. The control of the outcomes in the case study was reached by means of a set of preliminary simplified simulations; the assumption for each one is specified in Tab. 3. This step-by-step approach allowed to understand the effect of each geometry detail (curvature at the internal edge, lowering of the bottom of the intake channel etc.) on the flow field separately from the others.

	Lowering of the bottom	Curvature of the internal edge	Column into the canal
Case a)			
Case b)		X	X
Case c)	X		

Tab. 3. Geometrical elements included in the preliminary simulations

Fig. 2 shows the results of the three 3D simulations in terms of contour of velocity magnitude applied to a set of 3D streamlines released from the *inlet* surface. Any water recycle therefore is not appreciable.

The sharp edge in the case a) induces, as expected, the detachment of the streamlines from the lateral wall of the channel, resulting in a substantial contraction of the flow and in a formation of a turbulent zone, which propagates downstream up to the turbine impeller. The velocity magnitude pattern is typical of the sudden contraction case and reaches the maximum value in correspondence of the internal edge. Case b) shows the beneficial effect of the curved edge, which completely suppresses, in the simplified case of horizontal intake channel, the formation of recirculation zones. The central pier also induces a perturbation to the flow which, differently, does not reach the end of the channel, as shown by the well aligned streamlines. On the other hand, the presence of the pier affects the velocity distribution at the outlet of the channel. An important result comes with the case c) which shows the effect of the inclined bottom of the intake channel. In particular, the already presented effect of the contraction of the channel (case a)) becomes amplified, resulting in an extended recirculation zone that affect the water flow entering the turbine. We can conclude the intuitive result that both the curved edge and the column induce some perturbation to the flow and the less intuitive result that the inclined bottom strongly amplifies the two effects.

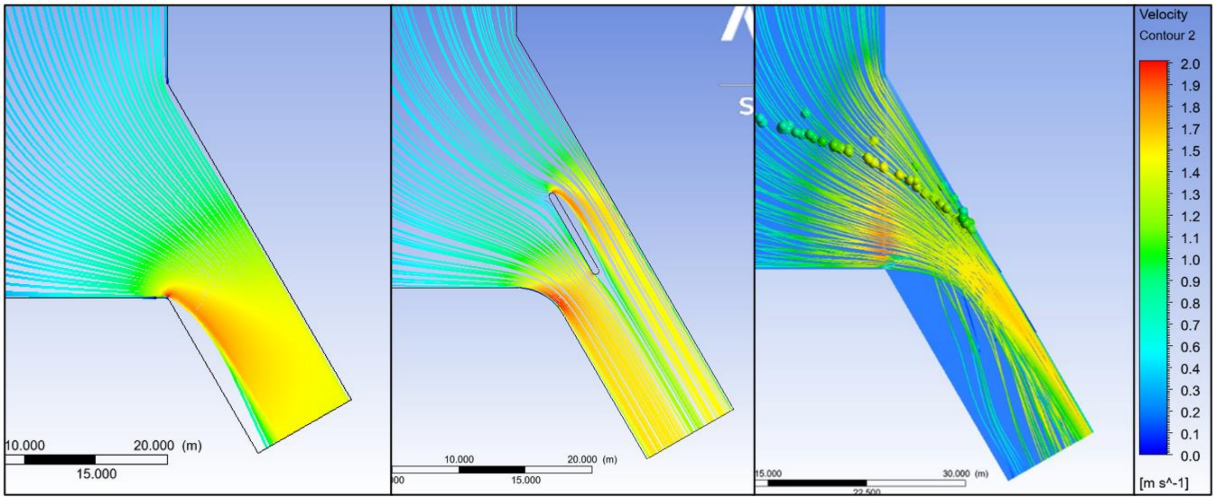

Fig. 2. 3D streamlines released from the inlet boundary – from the left to the right: case 1), case2), case 3)

2.3 CFD simulation of the designed plant

The final simulation included all the geometry details, studied separately in the previous chapter. The results of the simulation are shown in the following pictures. The computed free surface layer, shown in Fig. 3, at the almost constant level of 109.57 m a.s.l. – 5.58 m from the bottom of the channel (imposed at the outlet boundary). Fig. 4 shows the contour of velocity magnitude applied to:

- 3D streamlines released from the *inlet* (left)
- 2D streamlines visualized on three different horizontal planes located under the free surface at different depths (1 m, 1.5 m, 4 m from the free surface)

The results confirmed what already observed in the simplified preliminary simulations, hence, the effect of the curved edge and the one of the pier, both strongly amplificated by the lowering of the bottom of the intake channel. A more interesting outcome consists in the variation of the perturbation extension for different water depths (see 2D streamlines applied to different planes under the free surface, Fig. 4). Close to the left lateral wall, the superficial layer is interested by a strong perturbation which propagates downstream till the turbine intake, while close to the bottom of the channel the perturbation vanishes upstream of the turbine or is even absent. Similarly, the turbulence induced by the pier is stronger at superficial layers, while it is almost absent at higher depths. The velocity magnitude at the outlet boundary, shown in Fig. 3, is strongly non uniform, mainly due to the presence of the pier, which diverts the flow towards the left lateral wall of the channel. The values range between 0.47 m/s and 0.85 m/s. The plant layout, in its design configuration, revealed some criticisms which could result in a decrease of the turbine efficiency; therefore, an hydraulic optimization was needed.

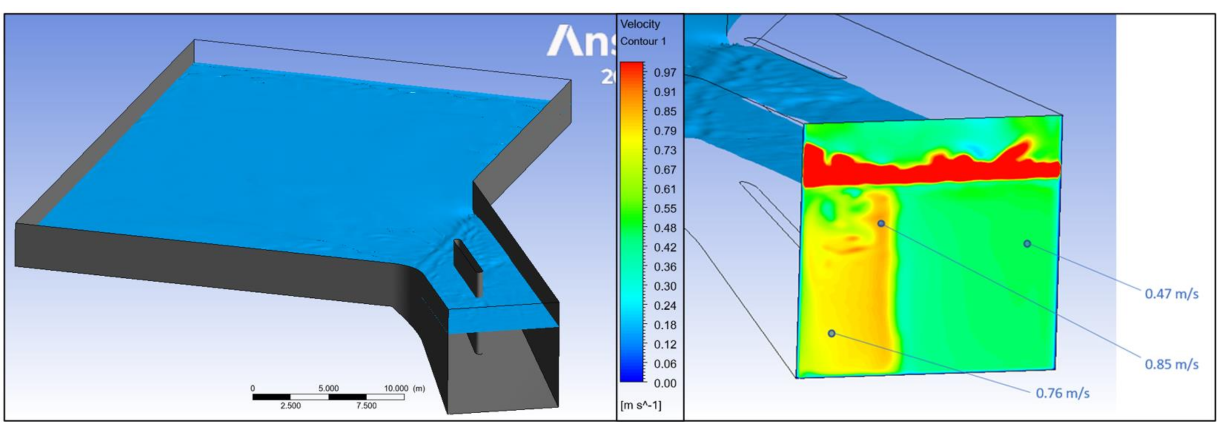

Fig. 3. Computed free surface layer (left) and velocity magnitude distribution at the outlet boundary (right)

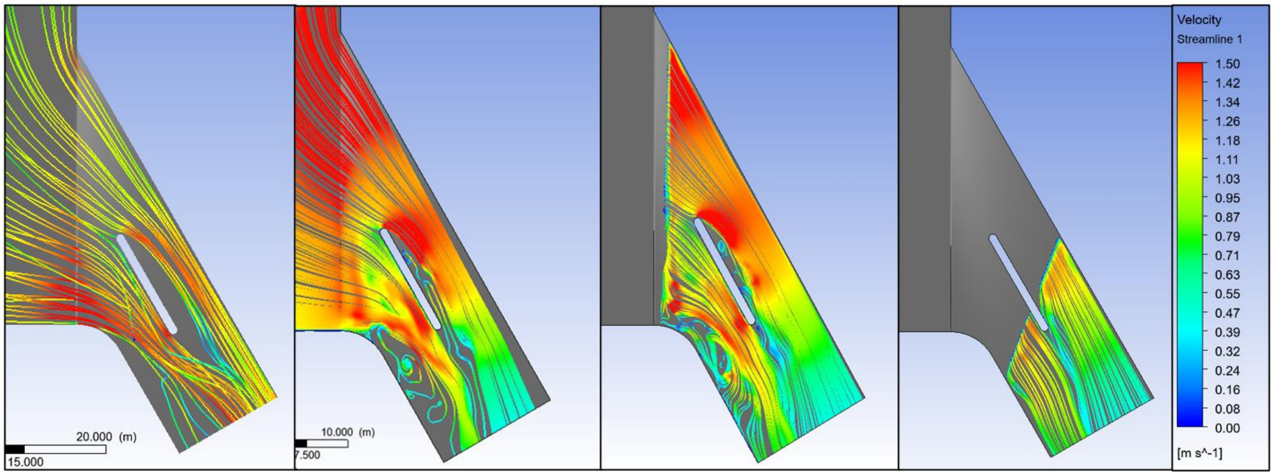

Fig. 4. 3D streamlines released from the inlet boundary (left) and 2D surface streamlines located 1 m, 1.5 and 4 meters under the free surface

3. Hydraulic optimization of the plant layout

Two different solutions have been considered:

a) Higher radius of curvature of the internal edge

The preliminary simulations, presented in chapter 2.2, highlighted the significant beneficial effect of providing a curved edge at the channel intake on the formation of turbulence. In this case the effect of an increase of the radius of curvature from the designed ~3 m to 5.7 m has been investigated. Fig. 5 shows, similarly to the previous cases, the visualization of 2D streamlines at the depths of 1 m and 3 m under the free surface and the velocity distribution at the *outlet* boundary. The resulting streamlines appear straight and well aligned all within the domain and no turbulent structures are present. In particular, the new curvature prevents the horizontal contraction of the flow, thus avoids the formation of recirculation zones close to the lateral wall. Furthermore, an overall redistribution of the flow velocities entering the channel intake can be noticed. The velocity of the flow hitting the pier is lower, resulting in a less perturbed flow. The velocity magnitude of the water at the outlet ranges between 0.60 m/s and 0.80 m/s and tends to be higher at one side of the surface because of the presence of the pier which, as already observed, diverts the incoming flow towards the lateral wall of the channel.

b) Opening of the environmental flow gate

Frosio Next experimentally observed, for a similar hydroelectric power plant, the beneficial effect induced by the partial opening of the environmental flow rate on the formation of turbulence at the channel intake. The solution has been considered for the current plant by assuming the presence of a rectangular gate located on the main weir and close to the channel contraction (see Fig. 6). Assuming the real water level upstream the weir (regulated from the turbine at 109.57 m a.s.l. – 1.5 m from the channel bottom), the environmental flow of about 2 m^3/s (design requirement) and a maximum allowable velocity through the gate of 1.5 m/s (admissible for concrete structures), it is needed a 2 meters width gate. During the solution computation it was monitored, together with the flow rate crossing the *output* boundary, the flow rate crossing the new opening, which stabilized after a certain time around the prescribed 2 m^3/s. The results of the simulation are shown below. The streamlines confirm, as expected, the beneficial effect of the gate opening on the formation of any turbulent structures, which are no more present. The velocity distribution at the outlet is similar to the one of the previous case and ranges between 0.64 m/s and 0.82 m/s.

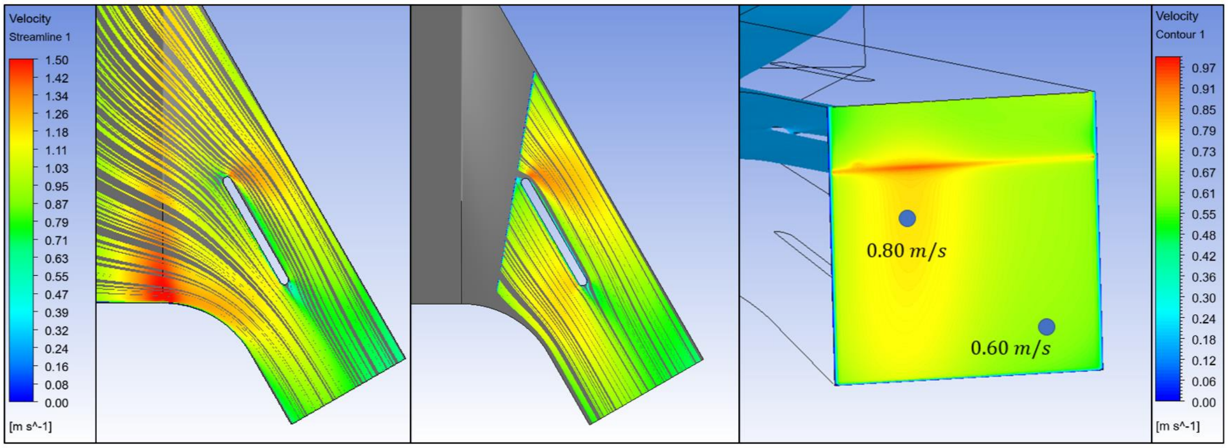

Fig. 5. 2D surface streamlines located 1 m and 3 m under the free surface (left) and velocity magnitude distribution at the outlet boundary (right) – case a)

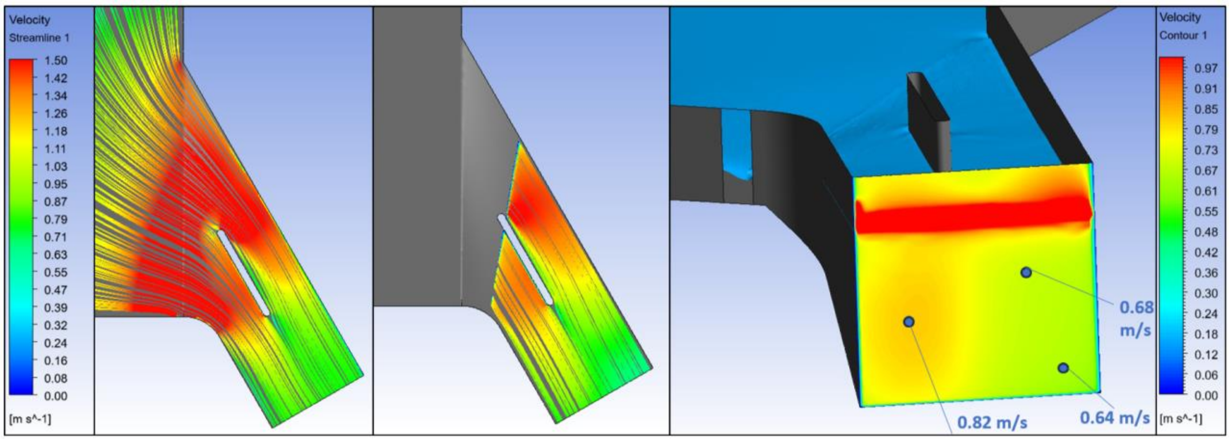

Fig. 6. 2D surface streamlines located 1 m and 3 m under the free surface (left) and velocity magnitude distribution at the outlet boundary (right) – case b)

4. Conclusions

The presented study highlighted the main design aspects to be taken into consideration for preventing turbulence within channel intakes of small run of river hydropower plants. The plant under investigation, in its design configuration, involves the formation of turbulent structures which generate inside the intake channel and propagate downstream up to the turbine, therefore impairing its efficiency. The main elements which play an essential role on the formation of such turbulence are the curved internal edge of the intake, the pier inside the intake channel, and the lowering of the channel bottom upstream the turbine. Keeping in place the local constraint of the case, hence the presence of the pier (needed to sustain a road) and of the inclined bottom (needed to host the turbine), the best solution for mitigating the formation of turbulence was found. Basically, it consists in improving the curved internal edge profile, by increasing its curvature. Reduction of turbulence may be also achievable through the installation of an environmental flow gate over the weir and close to the channel intake. This not intuitive outcome was confirmed from Frosio Next which observed experimentally the same phenomena on a similar power plant.
Evidence of turbulence phenomena in similar existing plants more and more justifies the choice of a CFD simulation in the design stage, especially in any case the plant layout differs from the optimal solution, due to, for example, the presence of local constraints or simply limited construction budget. In these cases, the main concern is guaranteeing the correct functioning of the plant without excessive loss of efficiency. On the other hand, the cost of a CFD simulation is always justified in case of high budget plants, where the cost for a CFD simulation becomes moderate with respect to the overall design cost. In these cases, different layout configurations may be explored in order to determine the one which maximise the efficiency of the plant.

The Authors

C. Fregoni: graduated in Civil and Environmental Engineering from the University of Brescia and joined Frosio Next (originally, Studio Frosio) in 2022. He has been involved in the hydraulic design related to hydroelectric power plants. He carried out hydrology analysis and held project manager roles. He conducted CFD simulations (2D and 3D) of open channel flows, especially channel intakes of low head plants for the determination of the high turbulence zones. He was involved in the evaluation of the conservation, efficiency, and functionality of several hydroelectric power plants in the northern Italy.

Luigi Lorenzo Papetti graduated in Hydraulic and Chemical Engineering from the University of Politecnico di Milano. Mr. Papetti is a long-time member of Frosio Next (formerly Studio Frosio) since 1990. Holding a M. Sc. degree in hydraulic engineering and being specialized in hydropower plants and hydraulic works, he is currently CEO and CTO of Frosio Next. During his career he gained a wide experience both in the refurbishment of existing plants and in the design of new ones. As ELMEC expert he prepared hundreds of technical specifications.